For cited: *Kütük, E., Çalışkan, M. & Bayram, A. (2018). A Study on Aerodynamic Drag of Semi Trailer Truck with Different Rear Height. Journal of Current Researches on Engineering, Science and Technology, 4 (2), 113-124.*

Journal of Current Researches on Engineering, Science and Technology (JoCREST)
ISSN: 2651-2521

Year: 2018
Volume: 4
Issue: 2

Crossref doi: 10.26579/jocrest-4.2.9

Research Article/Araştırma Makalesi

A Study on Aerodynamic Drag of Semi Trailer Truck with Different Rear Height

Ensar KÜTÜK[1], Mehmet ÇALIŞKAN[2] & Aykut BAYRAM[3]

Keywords
semi-trailer truck, aerodynamic, rear height, CFD, drag coefficient, Fluent®.

Article History
Received
21 Nov, 2018
Accepted
30 Dec, 2018

Abstract
Nowadays, with increased traffic of transport operations, fuel consumption and environmental impacts are becoming more important on heavy duty vehicles. This situation have leaded to many researches.
In this study, the aerodynamic drag of a semi-trailer truck with a reduced rear height according to the standard are compared and investigated. To measure the aerodynamic drag produced by the semi-trailer trucks, computational fluid dynamics (CFD) analysis study was undertaken using a 1/1 scale 3D model trucks. Ansys Fluent software has been used to analyze flow equations and to obtain the visual outputs.
The results were compared taking into account the payload volume and drag coefficient and in a scenario which internal volume is not important, it was observed that vehicle aerodynamic drag may decrease by up to 9.1% and resulting in a 4.7% improvement in fuel consumption.

1. Introduction

Heavy commercial vehicles are considered aerodynamically inefficient compared to other ground vehicles due to their un-streamlined body shapes. A large commercial vehicle traveling at 100 km/h consumes about approximately 52% of the total fuel to provide power to overcome the aerodynamic drag. (Dias et al., 2016) Road transportation is a main pollution emitter of CO2. About one-third of the CO2 emissions from road traffic are caused by commercial vehicles, especially from the widely used semi-trailer truck class. (Chowdhury et al., 2013) (Stadler et al., 2014)

Energy consumption and efficiency gain more importance every day in our world. As a result, many studies have been carried out and will carry out to reduce the energy used in transportation.

External aerodynamics simulations using computational fluid dynamics are well established tools in the product development process for the automotive and aerospace industries. (Dias et al., 2016)

[1] Corresponding Author, Mechanical Engineer, Tırsan Treyler A.Ş., ensar.kutuk@tirsan.com
[2] Assoc. Prof. Dr., Sakarya Uygulamalı Bilimler Üniversitesi, caliskan@sakarya.edu.tr
[3] Mechatronics Engineer, Tırsan Treyler A.Ş., aykut.bayram@tirsan.com

H. Chowdhury, H. Moria, A. Ali, I. Khan, F. Alam and S. Watkins (2013), had added including different external attachments (i.e., front faring, side skirting and gap filling) on the baseline vehicle. The results had shown that these external attachments have notable impact on aerodynamic drag as they can reduce up to around 26% aerodynamic drag over the baseline model.

In another study S. Stadler and M. Hirz (2014), examined the effect of aerodynamic drag of different rear shape types on the vehicles. As a result of the study, it is observed that the fuel consumed per unit load on some vehicles with different rear shapes is also good. Respectively, on vehicles which are interior volume changes by 3.2% and 6.1%, fuel consumption has declined by 6.5% and 10.2%. Also in this study, advices and included design work where the rear form of the vehicle can be adapted to the payload, had be published by writers.

In another study, C. Bayındırlı (2014), examined the aerodynamic recovery by adding a spoiler to the semi-trailer truck. With this add-on, had be got 20.9% improvement in C_D value at different speeds. In the study it was stated that this improvement corresponds to about 10% fuel consumption.

In another study, C. Bayındırlı, Y.E. Akansu and M.S. Salman (2016), had wanted to determine the aerodynamic coefficient of the scaled truck-semi trailer model by their wind tunnel experiments with different Reynolds numbers.

In another study, G. Dias, N.R. Tiwari, J.J. and Varghese, G.Koyeerath (2016), had reduced value of C_D as %6 with added rear spoiler.

In another study, C. Chilbule, A.Upadhyay and Y.Mukkamala (2014), had compared basic and modified semi-trailer trucks. The profile modification has been done on basic truck-trailer model by providing wind deflector on truck’s cabin, vortex trap, mini skirt, vortex strake and aerodynamic revolute. It has been analyzed that, due to profile modification the reduction in aerodynamic drag is up to 21 %, which reduces the fuel consumption by 4 liters for 100 km for diesel powered truck.

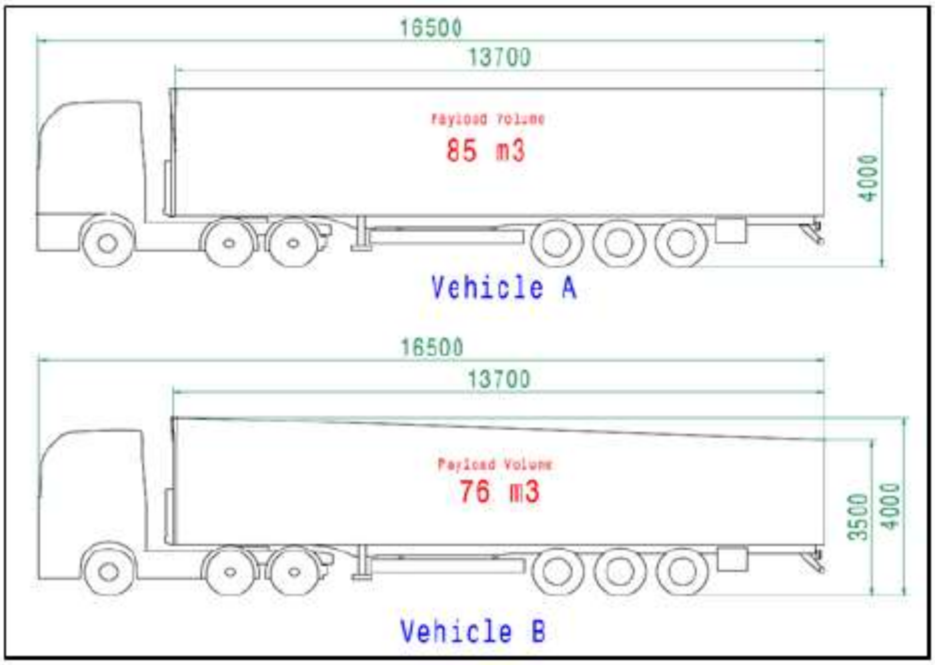
2. Objective

This paper focuses on the CFD aerodynamic analysis of 2 type semi-trailer truck vehicles. First type of vehicle (A) has standard dimensions truck and semi-trailer. The second one’s (B) rear height was reduced 500mm. (Fig.1)

With this improvement, it is aimed to reduce the turbulent flow behind the vehicle during driving on road.

The models of vehicles, created in Catia V5, is analyzed using analysis software Fluent-Ansys17. The goal is to simulate the air flow around this two vehicles and obtain an accurate value of its drag coefficient.

Figure 1. Type A and B vehicles dimensions and payload volumes



3. Theory

The aerodynamic drag force (F_D) on a vehicle is given as;

$$F_D = \frac{1}{2} C_d . A . \rho . V^2 \quad (1)$$

Where C_D is the non-dimensional drag coefficient, A is the frontal area, ρ is the density of surrounding air and V is the velocity of air with respect to the vehicle.

When the independent variable are taken into account, The C_D directly emerges as a result of the geometric shape of the vehicle. And the drag coefficient (C_D) for applications with complicated geometry can be extremely difficult to solve directly. For this reason the use of CFD software is quite common on solving this problems.

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics, in which the problems of fluid behavior are solved on the computer by numerical methods and algorithms. Software-side developments can easily model turbulent, multi-phase and supersonic flows.

The underlying equations that are solved in CFD problems are the Navier-Stokes equations. In the laminar regime, the flow of the fluid can be completely predicted by solving the steady-state Navier-Stokes equations, which predict the velocity and the pressure fields. However, the most common way to simulate turbulence is to calculate the time-averaged properties of the flow which in most cases give sufficient information about the flow. One of the suitable methods for this is The Reynolds Averaged Navier-Stokes (RANS). These equations can be used to give time averaged solutions to the Navier–Stokes equations. (Mangrulkar et al., 2014) (Hakansson et al., 2010)

3.1. Continuity Equation

The continuity equation is expressed as the mass balance in the control volume in a flow;

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad (2)$$

$$\frac{\partial\rho}{\partial t}+div(\rho\vec{u})=0 \quad (3)$$

$$div\vec{u}=0 \quad (4)$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (5)$$

3.2. Momentum Equation

According to Newton's second law, the rate of momentum change of a fluid particle is equal to the rate of change of the momentum of this fluid particle plus the forces acting on that fluid fraction. The momentum increase rate in the x, y and z directions of the unit volume of a fluid fraction is given by;

$$\rho\frac{Du}{Dt}\ ,\qquad \rho\frac{Dv}{Dt}\ ,\qquad \rho\frac{Dw}{Dt}$$

The x-component of the momentum equation;

$$\rho\frac{Du}{Dt}=\frac{\partial(-p+\tau_{xx})}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+S_{M_x} \quad (6)$$

$$\rho\frac{Dv}{Dt}=\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial(-p+\tau_{yy})}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+S_{M_y} \quad (7)$$

$$\rho\frac{Dw}{Dt}=+\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial(-p+\tau_{zz})}{\partial z}+S_{M_z} \quad (8)$$

3.3. Navier – Stokes Equation

Navier-Stokes and continuity equations are also referred to as differential motion equations. Some assumptions are taken in solving these equations. And three components (x, y, z) of pressure and velocity are calculated. (Bayındırlı, 2017),

The most useful way to develop the finite volume method of Navier-Stokes equations;

$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+div(\mu grad u)+S_{M_x} \quad (9)$$

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+div(\mu grad v)+S_{M_y} \quad (10)$$

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z}+div(\mu grad w)+S_{M_z} \quad (11)$$

3.4. RANS Equation

The RANS equations are as follows;

$$\frac{\partial}{\partial x}(u.A) = 0 \tag{12}$$

$$\rho\frac{D\bar{u}}{Dt} = -\frac{\partial\bar{p}}{\partial x} + \rho g_x + \frac{\partial}{\partial x}\left(\mu\frac{\partial\bar{u}}{\partial x} - \overline{\rho u'^2}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y} - \overline{\rho u'v'}\right) + \frac{\partial}{\partial z}\left(\mu\frac{\partial\bar{u}}{\partial z} - \overline{\rho u'w'}\right) \tag{13}$$

$$\rho\frac{D\bar{v}}{Dt} = -\frac{\partial\bar{p}}{\partial y} + \rho g_y + \frac{\partial}{\partial x}\left(\mu\frac{\partial\bar{v}}{\partial x} - \overline{\rho u'v'}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial\bar{v}}{\partial y} - \overline{\rho v'^2}\right) + \frac{\partial}{\partial z}\left(\mu\frac{\partial\bar{v}}{\partial z} - \overline{\rho u'w'}\right) \tag{14}$$

4. Models and Meshing

Vehicle combination models which used in study were prepared in Catia V5. Models have been prepared while being inspired and simplified from real models of vehicles. Behind they have been transferred to analysis environment with .iges file extension.

4.1. 3D Models of Vehicles

Figure 2. Basic and coupled model of Vehicle A

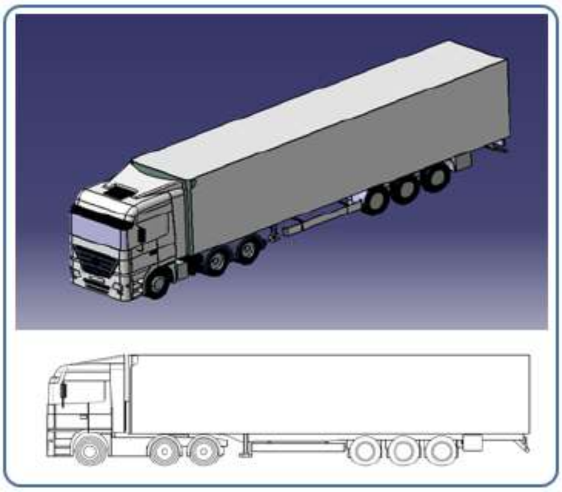
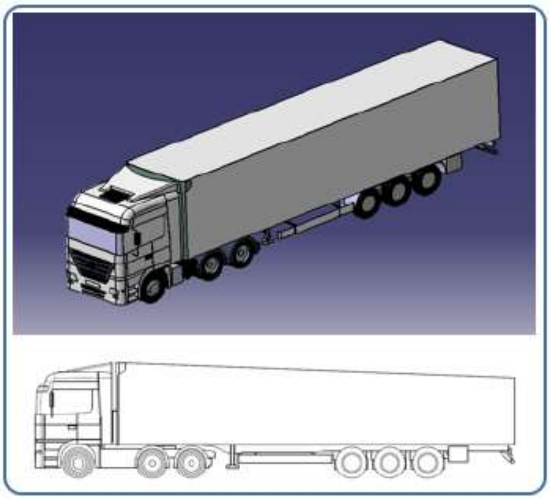


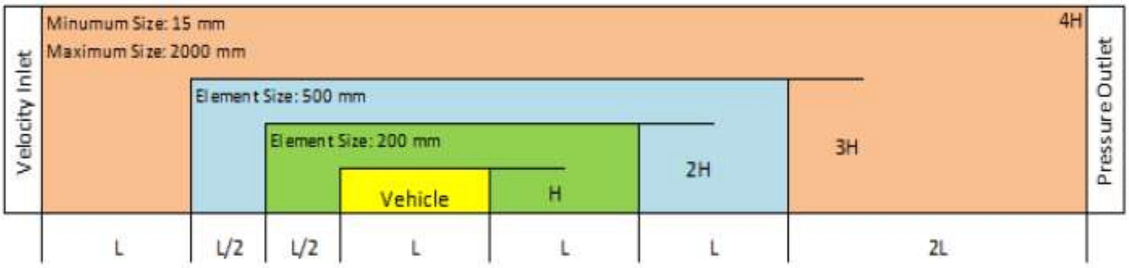
Figure 3. Basic and coupled model of Vehicle B



4.2. Mesh Structure

The previously determined criteria were used in the mesh modelers. (Fig.4) Mesh processing was performed with ansys mesh module for the study. The mesh number was increased in the vehicle surfaces area to increase the accuracy of the run-outs.

Figure 4. Mesh size parameters



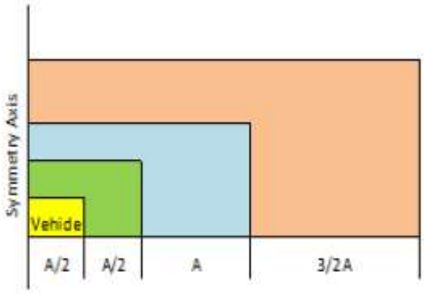
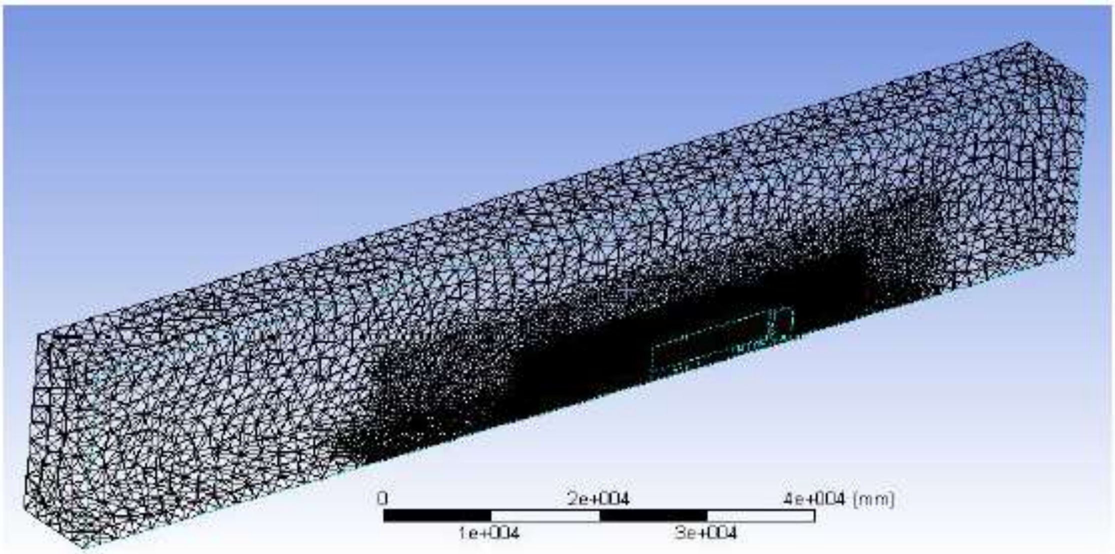


Figure 5. A image of Vehicle A mesh structure



5. Boundury Condition and Analysis Setup

In this study, Ansys Fluent software has been used to analyze flow equations and to obtain the visual outputs.

The boundary conditions and analysis setup parameters which are given as Table.1, Table.2, Table.3 and Table.4

Table 1. Solver setting

CFD Simulation	3D
Solver	Pressure-Based
Time	Steady
Velocity Formulation	Absolute

Table 2. Viscous model and turbulence model settings

Viscous Model	
Turbulence Model	k-ε (2 eqn)
k-epsilon Model	RNG
Near-Wall Treatment	Standart Wall Functions
Operating Conditions	Non-Equilibrium Wall Func.

Table 3. Boundary Condition

	Boundary Condition	
Velocity Inlet	Component	(-x) 100 km/h (constant)
	Specification Method	Intensity and Viscosity
	Turbulent Intensity %	1
	Viscosity Ratio	10
Pressure Outlet	Gauge Pressure magnitude	0
	Specification Method	Intensity and Viscosity
	Turbulent Intensity %	5
	Viscosity Ratio	10
Wall Zones	No Slip	
Symmetry	No Slip	
Fluid Proterties	Fluid Type	Air
	Density	P=1.225 (kg/m^3)
	Kinematic Viscosity	v=1.7894x10^{-5}(kg/(m•s))

Table 4. Interpolation options in numerical analysis of truck trailer

Pressure	Second order
Momentum	Second order upwind
Turbulent kinetic energy	First order upwind
Turbulent dissipation rate	First order upwind

6. Results

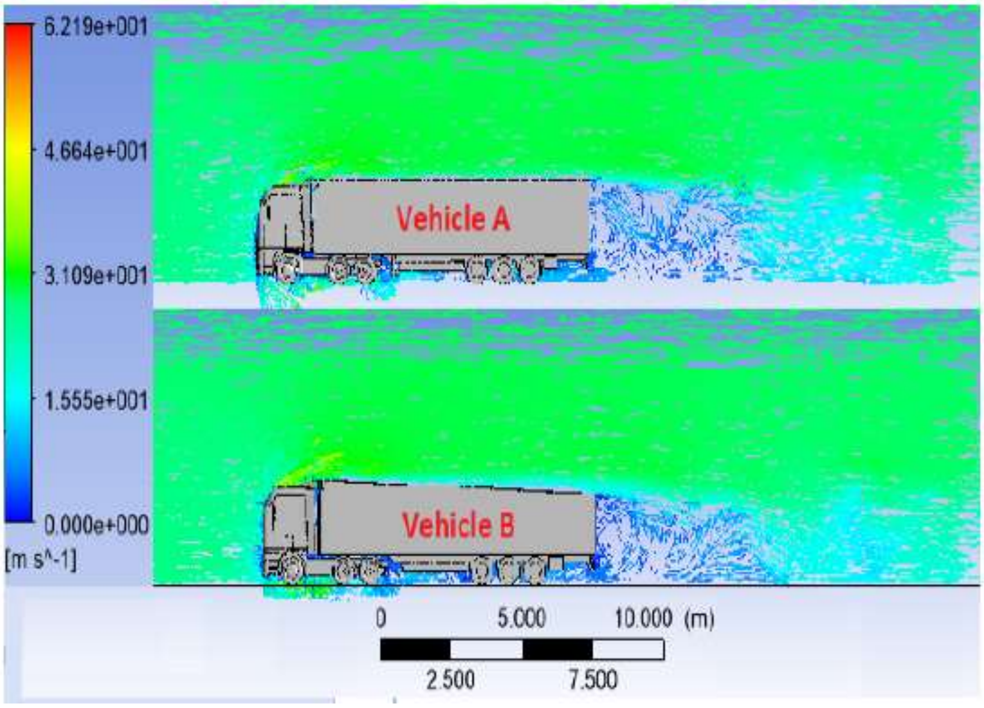
The iterations were carried up to 8.000 for reach the point where the change in the value of drag coefficient was found negligible for each model.

The results of the study in Table 5 are summarized. While calculating the gain in fuel consumption, it is recognized that in the semi-trailer trucks, 52% of the energy from the fuel is spent in aerodynamic losses, as indicated in the introduction.

Table 5. Comparison of drag coefficient, fuel consumption and payload space

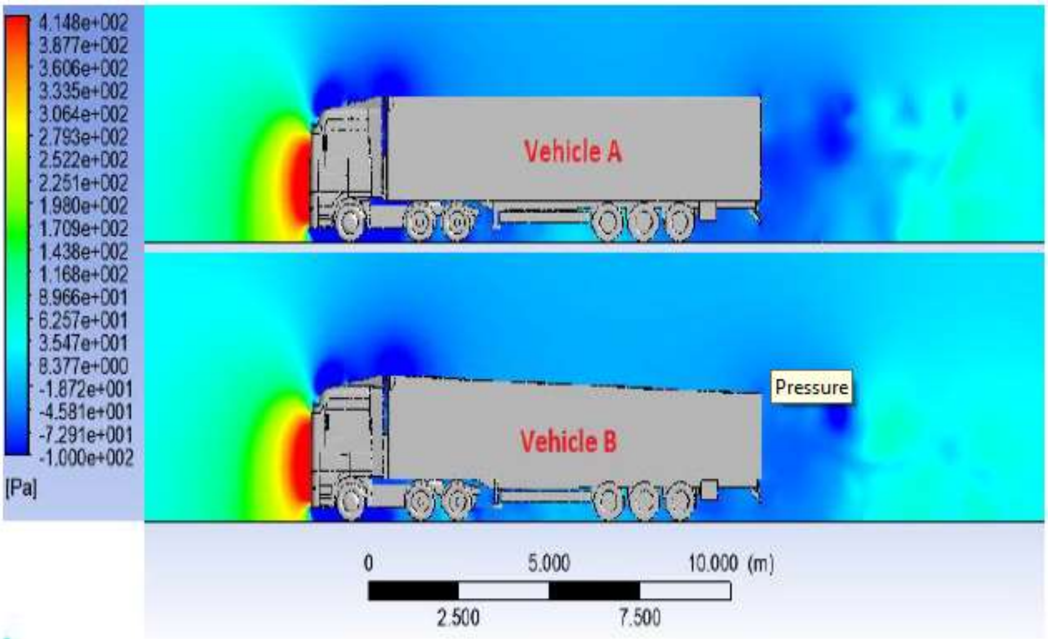
	Volume	C_D	Fuel Save
Vehicle A	85 m^3	0.464	
Vehicle B	76 m^3	0.422	≈ %4,7

Figure 6. Comparison of flow velocity vectors between Vehicle A and Vehicle B on 100km/h



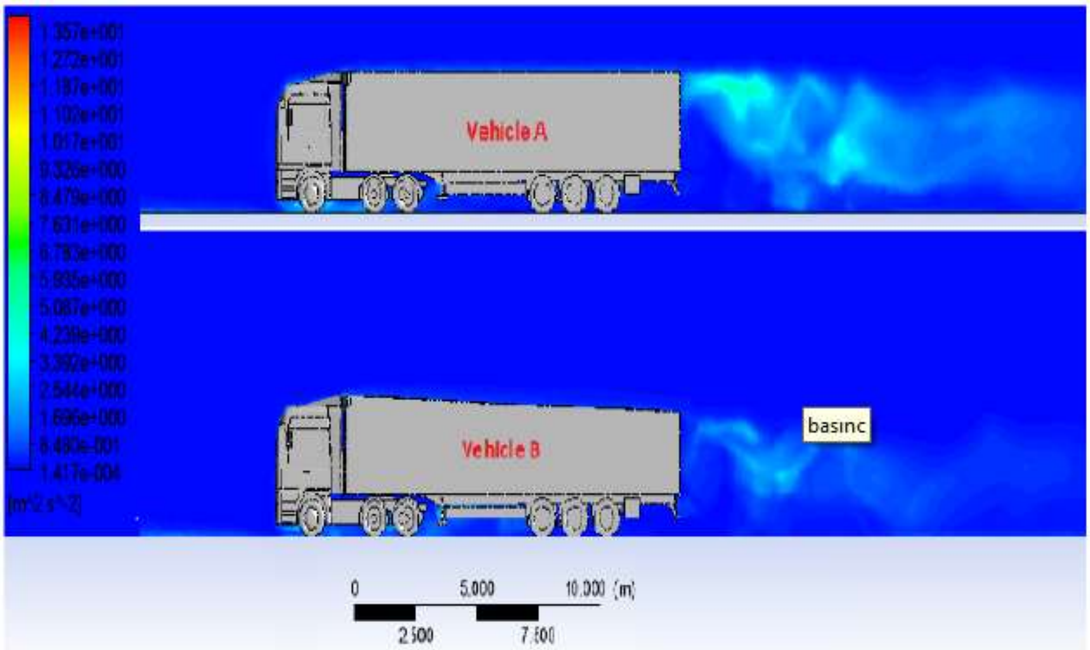
Both samples have high pressure areas at the front. When compared to Truck A and Truck B in the rear area, the magnitude and low pressure differential of the low pressure area in truck B has decreased greatly.

Figure 7. Comparison of pressure between Vehicle A and Vehicle B on 100km/h



As shown in Figure 8, there is a large turbulence field in the rear region of the truck. This situation is seen as a very regular flow regime in Truck B.

Figure 8. Comparison of flow turbulence between Vehicle A and Vehicle B on 100km/h



6. Discussion

In the study, a 9.1% improvement in C_D value was observed despite the 10% decrease in internal volume with the modification made in the vehicle. When compared to the similar study by S. Severin and M. Hirz (2014), values although the downward trend both in the end, is somewhat different in the results. In S.Severin's study, on vehicles which are interior volume changes by 3.2% and 6.1%, value of C_D has declined by 15% and 23% respectively. This efficiency may be caused by the reducing of the side walls end width of the vehicles in the similar study.

In addition, when compared other studies (Chowdhury-2013, Stadler-2014, Şahin-2008, Bayındırlı-2017, Bayındırlı-2016, Chilbule-2014), the C_D values in this study is somewhat lower than. But it is an admissible deviation and it may be due to the simplification of the models.

Vehicle bodies in the model had been drawn with sharp corners. If the sharp corners are given chamfered, the value of C_D and regime of flow could be a bit improved.

Taking into account the results of the study; It can be said that the reduction of the rear height of semi-tug trucks is a significant gain in fuel consumption. This application looks useful for low consumption. But at the same time, there are loading scenarios where volume is important in the heavy duty transport sector.

While this is the case, with an additional mechanism with low amortization time that allows both types of vehicles to be used during work, the vehicle's rear heights can be raised when desired and reduce when desired.

In this way, the fuel consumption can be reduced while the damage to the environment can be reduced to some extent.

Acknowledgments

This study is supported by Tırsan Treyler A.Ş. Research and Development Center.

References

Chowdhury, H., Moria, H., Ali, A., Khan, I., Alam, F. and Watkins, S. (2013), "A study on aerodynamic drag of a semi-trailer truck", Procedia Engineering, Vol.56 pp. 201-205.

Stadler, S. and Hirz, M. (2014), "A Noval Approach of Aerodynamic Optimization on Longdistance Transportation Trucks", FISITA World Congress 2014, 2-6 June, F2014MVC-025, Maastricht.

Şahin, C. (2008), "Ağır Yük Taşıtlarının Aerodinamik Şekil Direnç Katsayılarının Hesaplamalı Akışkanlar Mekaniği Yöntemi İle Analizi", İstanbul Teknik Üniversitesi Fen Bilimleri Enstitüsü, Master Thessis, İstanbul.

Mangrulkar, A., Parab, A., Sakarwala, A., Paste B., and Patil V. (2014),“Aerodynamic Analysis of a Car Model using Fluent-Ansys 14.5”, International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE), Vol.1 No.4, pp. 7-13.

Bayındırlı, C. (2017), “Çekici Römork Aracında Spoiler Yapısının Sürükleme Katsayısına Etkisinin Hesaplamalı Akışkanlar Mekaniği ile Analizi”, Journal of Polytechnic, Vol.20 No.2, pp. 251-256.

Dias, G., Tiwari, N.R., Varghese, J.J. and Koyeerath, G. (2016), “Aerodynamic Analysis of a Car for Reducing Drah Force”, IOSR Journal of Mechanical and Civil Engineering of Polytechnic, Vol.13, No.3 pp. 114-118.

Bayındırlı, C., Akansu, Y.E. and Salman, M.S. (2016), “The Determination of Aerodynamic Drag Coefficient of Truck and Trailer Model by Wind Tunnel Tests”, International Journal of Automotive Engineering and Technologies, Vol.5, No.2 pp.53-60.

Chilbule, C., Upadhyay A. and Mukkamala, Y. (2014), “Analyzing the profile modification of truck-trailer to prune the aerodynamic drag and its repercussion on fuel consumption”, Procedia Engineering, Vol.97, pp.201-205.

Ansari, A.R. (2017), “CFD Analysis of Aerodynamic Design of Tata Indica Car”, International Journal of Mechanical Engineering and Technology, Vol.8, No.3 pp.344-355.

Siva, G. and Laganathan, V. (2016), “Design and Aerodynamic Analysis of a Car to Improve Performance”, Middle-East Journal of Scientific Research, Vol.24, pp.133-140.

Hakansson, C. and Lenngren, M. J. (2010), “CFD Analysis of Aerodynamic Trailer Devices for Drag Reduction of Heavy Duty Trucks”, Chalmers University of Technology, Master Thessis – Göteborg, 2010